

**Improved Inhomogeneous Finite Elements for
Fabric Reinforced Composite Mechanics Analysis**

R. L. Foye

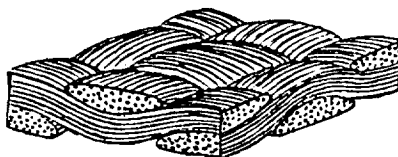
**North Carolina A&T State University
Greensboro, NC**

**Lockheed Engineering & Sciences
Hampton, VA**

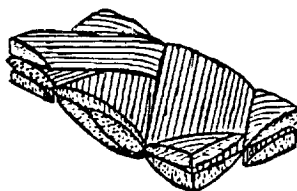
Introduction

Inhomogeneous finite elements are an attractive alternative to homogeneous elements in the mechanical analysis of fabric reinforced composites (Figure 1). These elements greatly simplify the mesh generation problem created by the complex reinforcing geometry. However, this advantage has some drawbacks associated with it. Convergence, with diminishing element size, becomes less certain. Also, the computation of stresses within the various constituent materials of an element becomes a problem. This paper addresses both these concerns.

The convergence can be improved by replacing the inhomogeneous elements with special homogeneous elements whose properties are chosen to match the inhomogeneous element response to simple average strain states. One and two dimensional examples are considered. The three dimensional application is discussed without examples. The analysis also provides the basis for an approximate solution for the average stresses within each constituent material of each inhomogeneous element. This makes an approximate stress/failure analysis possible.



PLAIN WEAVE



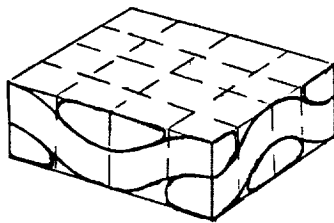
2 X 2 BRAID WITH STUFFER

FIGURE 1. FABRIC REINFORCED COMPOSITE MICROGEOMETRIES

The Analysis

The analysis uses the unit cell concept to create a boundary value problem that fully characterizes the reinforcing microgeometry. Division into subcells then establishes a finite element model of the unit cell and reduces the reinforcing complexity in each subcell to the point where average, subcell, constituent stress levels are meaningful. For example, consider the plain weave unit cell in Figure 2. Subcells that were adequate for stiffness analysis must be reduced in size to yield meaningful detailed stress information.

The individual subcell stiffness matrix, $[k]$, can be obtained by numerical integration of the general finite element energy formula $[k] = \iiint B^T D B \, dvol$ where the matrix D contains only material property distribution functions and the matrix B contains only displacement mode shape derivatives (Ref. 1). Once a library of different subcell stiffness matrices has been created it remains to transform them into the global coordinates and assemble them into an overall stiffness matrix for the unit cell. The surface nodal forces and the average strains in each subcell, corresponding to each of the six independent strain states of 3-D elasticity, may then be solved. The surface nodal forces give the average stresses on the unit cell surfaces. The stiffness coefficients of the composite may then be computed. The convergence of this process is considered first. The subsequent computation of the constituent stresses is taken up later.



UNIT
CELL



SUBCELL FOR
STIFFNESS ANALYSIS



SUBCELL FOR
STRESS ANALYSIS

FIGURE 2. DIVISIONS OF UNIT CELL

One Dimensional Example

The tension bar made from dissimilar materials (Figure 3) illustrates the convergence problems associated with inhomogeneous elements and simple displacement mode shapes. If a finite element node coincides with the point of material discontinuity then the elements become homogeneous, the strain in each element becomes uniform, and the analysis converges abruptly to the true displacements. But, if the material discontinuity is always contained within some element, as in Figure 3, then the solution is approximate and the accuracy and convergence rate depend on the choice of assumed displacement mode shapes. For example, consider a linearly varying displacement mode shape within each element and an internal node placement at the 1/3 and 2/3 points along the bar. Each subsequent refinement divides each element into three equal segments. The middle element remains inhomogeneous as element size decreases. Figure 3 is a plot of bar elongation error as the element size diminishes. The load point displacement approaches the exact solution monotonically. Although rapid, this convergence is less satisfactory than that of the homogeneous element solution.

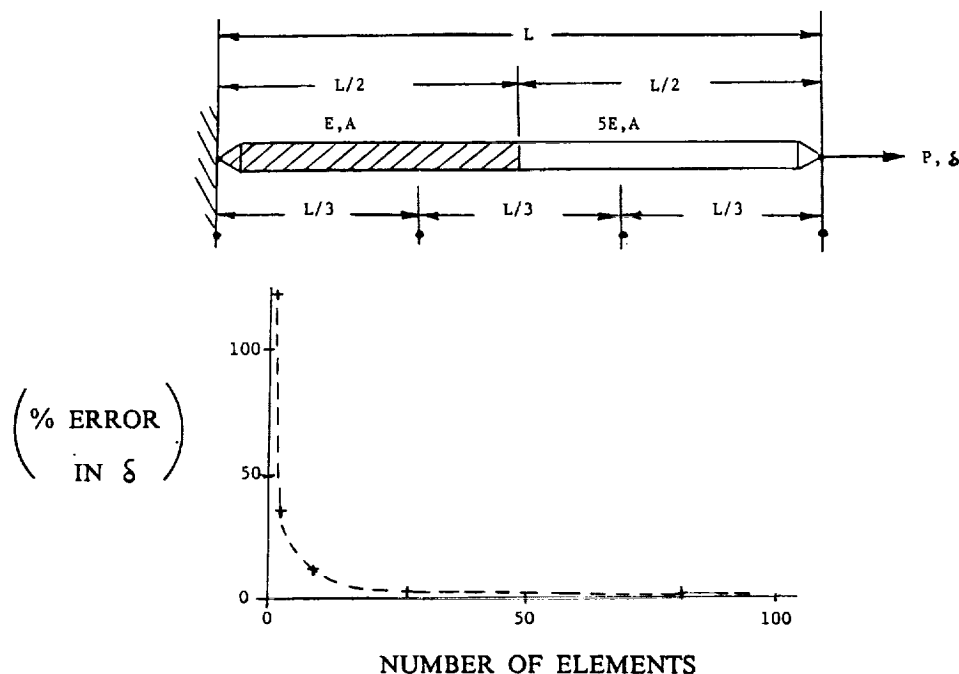
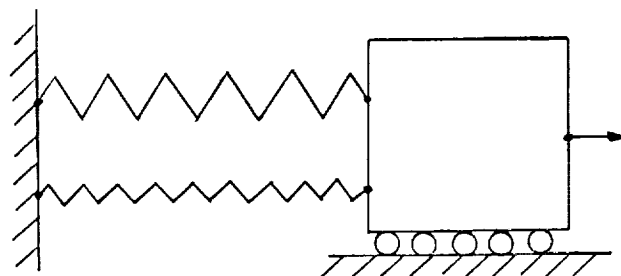


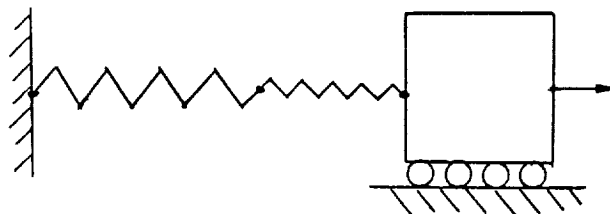
FIGURE 3. TENSION BAR PROBLEM

Improved Convergence

The convergence rate for the previous problem can be improved by using higher order displacement mode shapes (Ref. 2). It can also be improved by making a modification that overrides the source of error. The error source is the inability of the energy formulation, in combination with the linear displacement assumption, to distinguish between dissimilar material stiffnesses in series and parallel (Figure 4). The use of low order displacement modes is a presumption of parallel response when in reality the stiffnesses are functioning in series, in this particular application at least. The use of higher order modes would permit the analysis to make such a distinction. The same refinement can be attained by intervention of the analyst. Since the two materials in the center element are truly arranged in series, an effective modulus, E , of this element can be computed from the elementary series formulation $\frac{2}{E} = \frac{1}{E_L} + \frac{1}{E_R}$ where subscripts L, R designate left and right halves of the bar. The inhomogeneous center element can then be replaced by a homogeneous one with the modulus E . This correction leads to the immediate convergence of the deflection analysis. The logic in this argument seems trivial but as the dimension increases to two and three it becomes more complicated.



ELEMENTS IN PARALLEL



ELEMENTS IN SERIES

FIGURE 4. ELEMENTS IN SERIES AND PARALLEL

2-D Example

Figure 5 contains an example of a bimetallic composite in the form of bonded sheets of dissimilar materials. In the natural coordinates of the material the composite stiffnesses and internal stresses can be established from elementary mechanics. The results can be transformed into any other coordinates. In the coordinate system of Figure 5A the material can also be analyzed using inhomogeneous finite elements. A unit cell and a subcell division are shown in Figure 5B. Using the displacement modes usually associated with 8 node-isoparametric-brick elements (Ref. 1), generalized plane strain analysis, and the 32 node finite elements grid shown; the analysis overestimates the x and y moduli by over 20%. Refinement of the finite element grid leads to the composite moduli estimates of Figure 5C.

The convergence may be improved by recognizing how the reinforcing layers provide stiffness and then applying the appropriate rule of mixtures formulae while substituting homogeneous orthotropic elements in place of the inhomogeneous ones. Stiffnesses in parallel apply in the two principal reinforcing directions. Stiffnesses in series apply normal to the material boundaries. Figure 5C shows that this leads to abrupt convergence of the finite element sequence. The Appendix contains the relevant equations for this analysis.

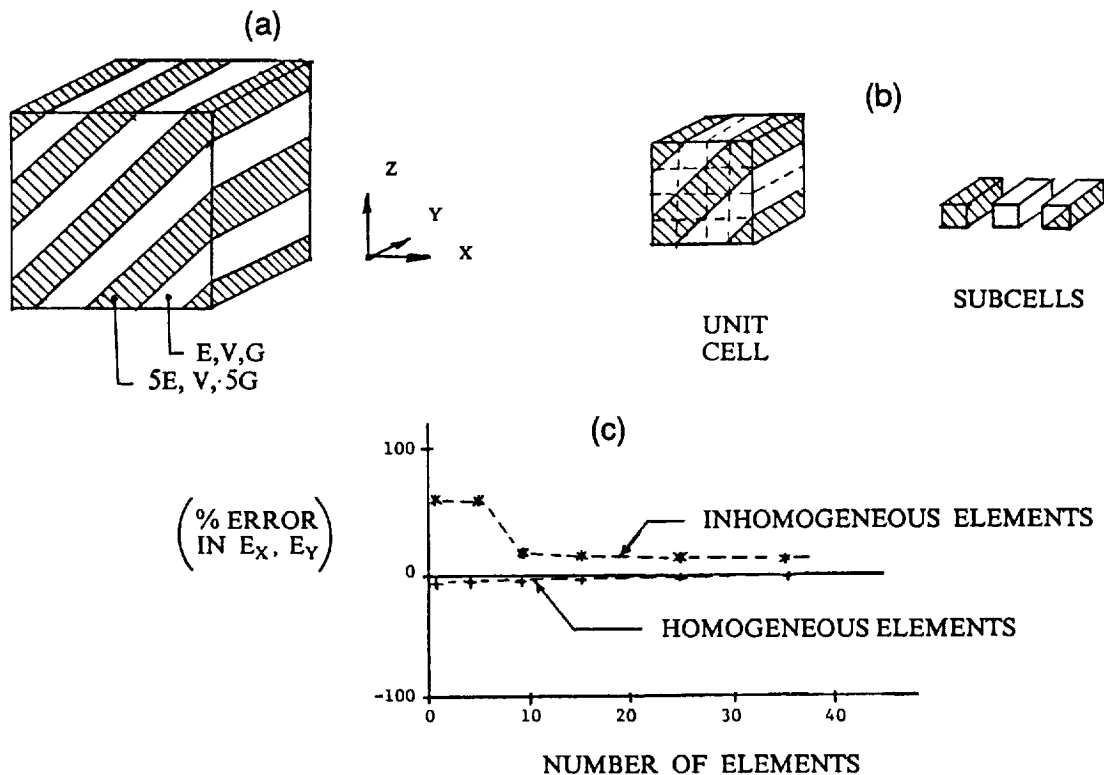


FIGURE 5. LAMINATED 2-D COMPOSITE MATERIAL

2-D Generalization

Consider generalizing the 2-D case to include more than two materials while restricting the material distribution within the subcell. Assume the material boundaries can be represented by a number of straight lines radiating from a common point within the subcell (Figure 6A). If additional lines are drawn from that point to each subcell corner node then each material will be contained in one or more homogeneous triangular finite elements. Corner displacements of the rectangular subcell are fixed by the applied strain case. Edge conditions must be approximated. The stresses within each material of the subcell can now be computed corresponding to each independent unit strain case. The nodal forces along the sides of the subcell are also available for calculating the mean stress/strain relations for the subcell. The subcell can then be considered homogeneous and anisotropic in the unit cell analysis.

If the common point lies outside the subcell boundaries then a similar analysis can be performed using some trapezoidal elements (Figure 6B). In either case full displacement continuity is not preserved between adjacent rectangular subcells.

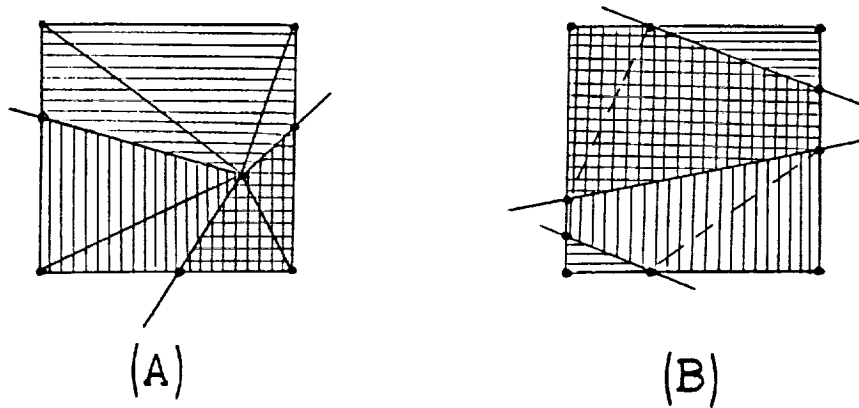


FIGURE 6. INHOMOGENEOUS/HOMOGENEOUS ELEMENT REPLACEMENT

Edge Boundary Conditions

The aforementioned boundary conditions on nodes created by the intersection of material boundaries and subcell edges are key points (Figure 7A). The assumptions used here are motivated by consideration of the special case where a material boundary and subcell edge cross at right angles (Figure 7B) and the surrounding material/loading arrangement is symmetric across the same subcell edge. Then the response will be symmetric. This implies zero tangential displacement along the material boundary (except for rigid motion) and zero nodal force normal to the material boundary.

The unit shear strain case (Figure 7C), relative to the same axes, represents antisymmetric loading with zero displacement normal to the material boundary (except for rigid motion) and zero nodal force component parallel to the material boundary.

For material boundaries which intersect subcell edges at angles other than 90° the true boundary conditions cannot be rationalized so easily. Nevertheless, it is assumed that the same displacement and force conditions remain as suitable approximations to the true conditions of equilibrium and compatibility across adjacent subcell boundaries.

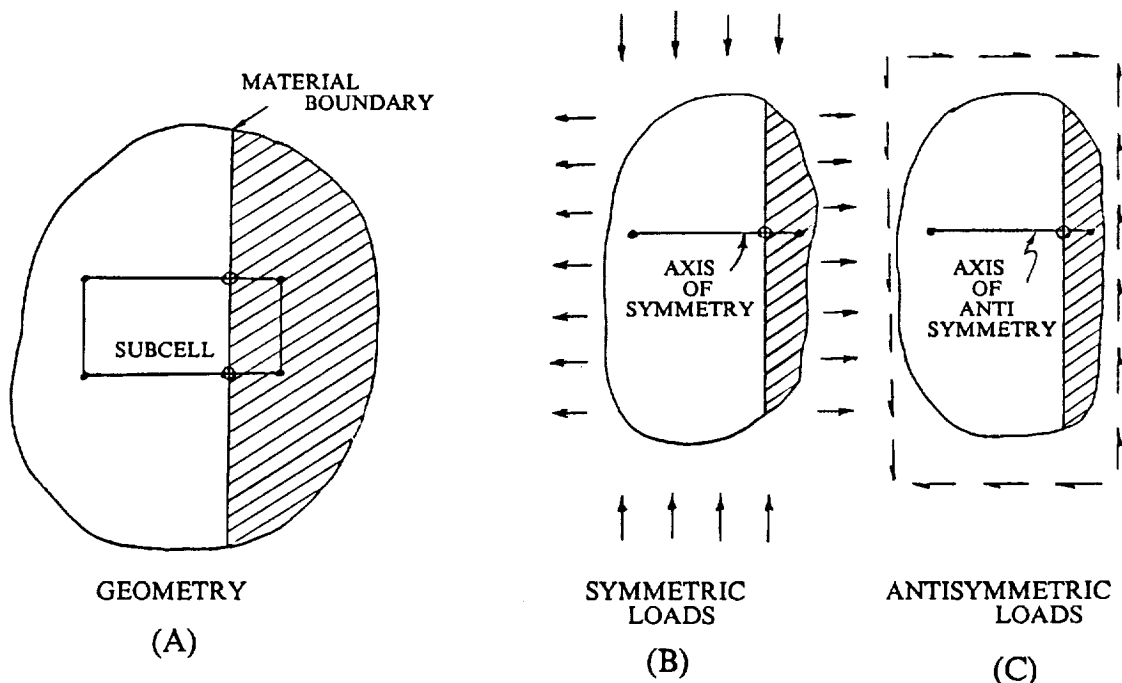


FIGURE 7. INTERFACE BOUNDARY CONDITIONS

3-D Analysis

To extend the analysis into three dimensions it is assumed that within any subcell of the rectangular 3-D array of subcells representing the unit cell, the dissimilar material boundaries consist of planes which intersect at a single straight line within the subcell (Figure 8). • Call this line the "common material axis". Consider the smallest sphere containing the subcell. A major circumference of the sphere exists which defines a plane which is normal to the common material axis and passes through the subcell centroid. By assuming the stresses do not vary along the common material axis the analysis reduces to a series of 2-D problems in the circumferential plane. Each 2-D problem resolves the displacements as a result of an applied unit strain with reference to the circumferential plane. The stresses within each wedge of dissimilar material may then be solved and the average stresses over any planar area within the subcell computed. These average stresses over the subcell surfaces permit the calculation of the equivalent anisotropic constants of the subcell. The boundary conditions at node points formed by the intersection of dissimilar material planes and subcell boundaries are established in the same way as in the 2-D case. Thus, inhomogeneous subcells are replaced by homogeneous ones in the unit cell analysis.

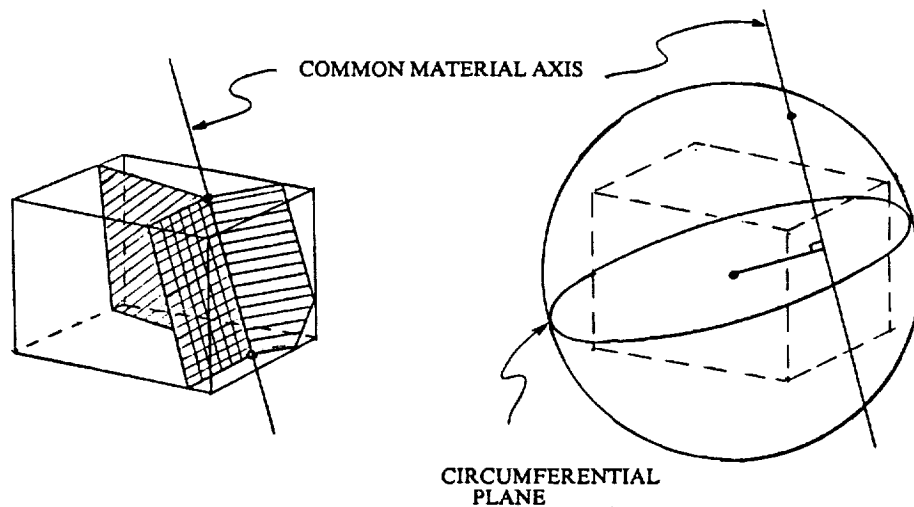


FIGURE 8. 3-D INHOMOGENEOUS SUBCELL GEOMETRY

Global-Local Analysis

It is likely that critical regions of high stress within a unit cell can be identified beforehand, making a complete stress analysis unnecessary. Certain subcells or portions of subcells will be of interest and can be isolated for detailed study through global-local analysis. This reduces the computational effort. For example, a crude model of a unit cell, based on a few subcells, can be used to obtain composite stiffnesses and define the displacements on the boundaries of a smaller volume of microstructure that contains a single subcell of further interest. A more refined grid may then be superposed on the smaller volume and more detailed stresses or displacements obtained for either a failure analysis or a more refined analysis of some smaller portion of that volume (Figure 9). Such a sequence of grid refinement may be expected to yield detailed finite element average stresses and strains in regions of tow bypass, tow contract, and sharp tow curvature. The problem of resolving these average element strains into average constituent strains and stresses will be considered next.

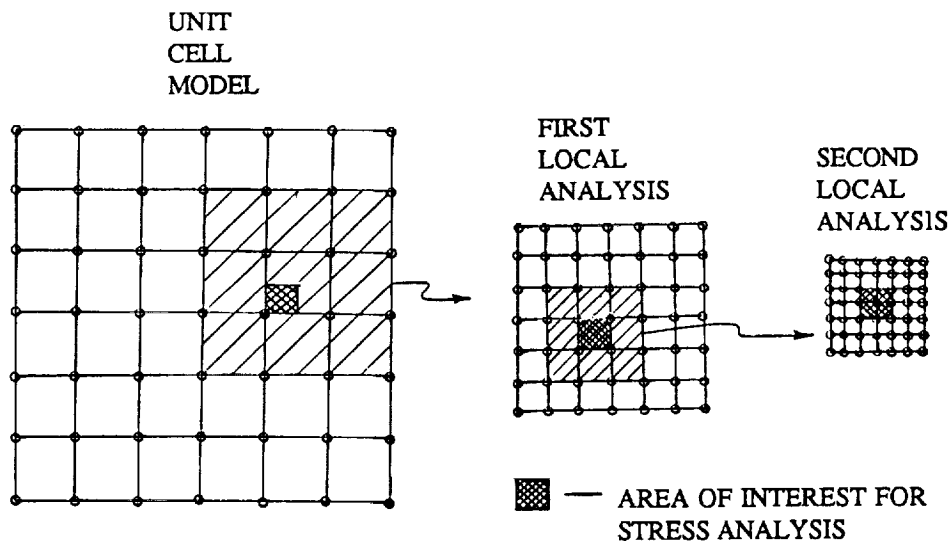


FIGURE 9. TYPICAL SEQUENCE OF INCREASING ANALYSIS DETAIL

Constituent Stresses

Returning to the one dimensional example, the determination of the stresses in the truly homogeneous elements of the tension bar is the same as in any finite element analysis. The strain is first computed from the displacements of the element nodes. The homogeneous, uniaxial stress/strain law of the material then leads to a stress calculation. For an inhomogeneous element the average strains are computed in the same way. Recourse to the stiffness in the series model that was used to compute the equivalent stiffness, E , also provides the basis for the constituent material stress calculation. The equations of strain compatibility and stress equilibrium plus the individual constituent material stress/strain relations are adequate in number for the stress calculations (Ref. 2).

In the general 2-D and 3-D cases the same mini-finite-element models that form a basis for equivalent stiffness calculations also provide a mechanism for backing out the constituent stresses (Figure 10). First, the element corner displacements, in conjunction with the displacement mode shapes establish internal displacements. Derivatives of the mode shapes establish detailed strains. The volume averages of these detailed strains establish the subcell average strains. Each average strain component, along with the mini-finite element model yields average strains in each constituent material. Constituent stress/strain laws then yield average stresses at the constituent material level.

SELECT A RECTANGULAR UNIT CELL
SUPERPOSE RECTANGULAR FINITE ELEMENT GRID ON UNIT CELL
REPLACE INHOMOGENEOUS ELEMENTS WITH MIMICKING HOMOGENEOUS ONES
(VIA MINI FINITE ELEMENT MODEL)
ANALYZE UNIT CELL FOR EACH OF THE SIX INDEPENDENT UNIT STRAIN CASES
COMPUTE COMPOSITE MODULI
COMPUTE UNIT CELL STRAINS FOR APPLIED LOAD CASES OF INTEREST
RATIO CONSTITUENT STRESSES IN EACH ELEMENT FROM UNIT STRAIN CASES TO
APPLIED LOAD CASES
COMBINE CONSTITUENT STRESSES FOR THE APPLIED LOAD CASE STRAINS

FIGURE 10. STEPS IN STRESS ANALYSIS

Concluding Remarks

There is a need to do routine stress/failure analysis of fabric reinforced composite microstructures to provide additional confidence in critical applications and guide materials development. Conventional methods of 3-D stress analysis are time consuming to set up, run and interpret. A need exists for simpler methods of modeling these structures and analyzing the models. The principal difficulty is the discrete element mesh generation problem. Inhomogeneous finite elements are worth investigating for application to these problems because they eliminate the mesh generation problem. However, there are penalties associated with these elements. Their convergence rates can be slow compared to homogeneous elements. Also, there is no accepted method for obtaining detailed stresses in the constituent materials of each element. This paper shows that the convergence rate can be significantly improved by a simple device which substitutes homogeneous elements for the inhomogeneous ones. The device is shown to work well in simple one and two dimensional problems. However, demonstration of the application to more complex two and three dimensional problems remains to be done. Work is also progressing toward more realistic fabric microstructural geometries.

FABRIC MICROSTRUCTURAL ANALYSIS NEEDED

CONVENTIONAL (HOMOGENEOUS) FINITE ELEMENTS DIFFICULT TO APPLY

TO FABRIC

SIMPLER METHODS/MODELS NEEDED

INHOMOGENEOUS ELEMENTS ATTRACTIVE ALTERNATIVE

SLOW CONVERGENCE PROBLEM CAN BE REMEDIED

CONSTITUENT MATERIAL STRESSES OBTAINABLE

ANALYSIS NOT PROVEN YET

WORK REMAINS TO BE DONE

FIGURE 11. CONCLUSIONS

List of References

1. Foye, R. L. "The Mechanics of Fabric Reinforced Composites", Fiber-Tex 1988, NASA Conference Publication 3038, 1989.
2. Dellinger, G. and Foye, R.L. "An Approximate Method of Stress Analysis for Fabric Reinforced Composites", Fiber-Tex 1989, NASA Conference Publication 3082, 1990.
3. Foye, R. L., The Transverse Poisson's Ratio Estimate for Unidirectional Composite Materials, Journal of Composite Materials, May. 1972.

Appendix

The rectangular inhomogeneous element of Figure 12 consists of two homogeneous isotropic materials (A and B). Parallel to the material boundary plane ($\bar{y} = 0$) the average Young's moduli ($E_{\bar{x}}, E_{\bar{z}}$) and Poisson's Ratios ($\nu_{\bar{x}\bar{y}}, \nu_{\bar{y}\bar{z}}$) can be approximated by the parallel Rule of Mixtures:

$$E_{\bar{x}} = E_{\bar{z}} = E_A v_A + E_B v_B, \quad \nu_{\bar{x}\bar{y}} = \nu_{\bar{y}\bar{z}} = \nu_A v_A + \nu_B v_B$$

where v designates material volume fraction and subscripts A, B designate the material. The shear modulus ($G_{\bar{x}\bar{z}}$) can be approximated by the same rule

$$G_{\bar{x}\bar{z}} = G_A v_A + G_B v_B$$

Normal to the material boundary plane the average Young's modulus ($E_{\bar{y}}$) and shear modulus ($G_{\bar{x}\bar{y}}$) are given by the series Rule of Mixtures:

$$\frac{1}{E_{\bar{y}}} = \frac{v_A}{E_A} + \frac{v_B}{E_B}, \quad \frac{1}{G_{\bar{x}\bar{y}}} = \frac{v_A}{G_A} + \frac{v_B}{G_B}$$

Considering $\bar{y} = 0$ to be a plane of isotropy, the Poisson's Ratio ($\nu_{\bar{x}\bar{z}}$ or $\nu_{\bar{z}\bar{x}}$) in that plane can be approximated by (Ref. 3):

$$\nu_{\bar{x}\bar{z}} = \nu_{\bar{z}\bar{x}} = \frac{E_{\bar{z}}}{E_A} \left\{ v_A - (1-v_A) \left[\frac{E_B v_A - E_A v_B}{E_B (1-v_A) + E_A (1-v_B) R} \right] \right\} \quad \text{where } R = \frac{v_A}{v_B}$$

These equations are sufficient to support the construction of a homogeneous finite element stiffness matrix after the moduli are transformed into the global x, y, z coordinates.

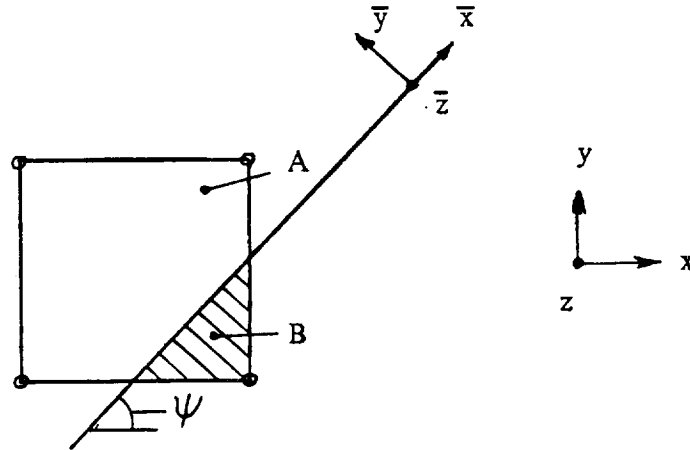


Figure 12: 2-D Analysis for Two Materials

